Main activities of the MSNM-GP laboratory

Sébastien Poncet, Patrick Bontoux, Guido Ciraolo, Dominique Morvan, Éric Serre
Laboratoire MSNM-GP, UMR 6181 Universités d'Aix-Marseille,
Technopôle Château-Gombert, 38 rue Joliot-Curie, 13451 Marseille, France

1 Introduction

In January 2008, the MSNM-GP laboratory will be composed of about 54 permanent persons - corresponding to 46 researchers and faculty members, 8 staff and engineers - plus about 30 PhD students and post-doctorates. The laboratory is organized in two Departments - Computational Fluid Mechanics and of Chemical engineering - that cover nearly 60% and 40%, respectively. It is mainly located in two sites in the campus of Ecole Centrale de Marseille - IMT Château-Gombert - and in the campus of Arbois - north of Marseille. The MSNM-GP laboratory covers a wide range of topics mainly relevant to fluids dynamics and transfer, of high order accurate approximations and of high performance computing.

The activities of the group of heat transfer and reactive flows involves topics as the modelling and physical investigation of instabilities in combustion [10], the simulation of fire expansion in natural (forest) sites [6, 7] and the combined numerical and experimental (with industrial companies) study of smoke expansion and heat transfer in tunnels during fires. Many topics concern the understanding of fluid flow and heat transfer processes in microgravity environment (CNES) - as for instance, the hydrodynamics of supercritical fluids (pure CO2 or binary mixtures [8]) and flow regimes, the convection and thermal distribution of species inside a closed cabin [4] ... The supercritical fluid flow topic [1] has also an impact in the control of processes as crystallisation inside solvents included in processes in junction with one of the team of the Chemical engineering Department (ESA program) [9]. This (mainly experiments oriented) department also involves teams acting in the membrane and filtration processes and in the treatment of waste water and it has an important role in the collaborative work with industries.

The Fluid Mechanics Department is involved in the modelling and simulation of instabilities, transitional and turbulent flows in internal and external devices. The more recent contributions concern the investigation of the vortex breakdown in a cylinder with a rotating bottom and a free surface [13] and also the investigation of LES based on special modelling issued for the spectral vanishing viscosity model [15] and using a penalization method (in strong link with the J.A. Dieudonné laboratory, R. Pasquetti) and Tech. Uni. Darmstadt (frame GDR Européen MFN, CNRS-DFG, M. Schäfer). The Coherent Vortex Simulation CVS has been extended to large size computational problem and the wavelet approach was developped in the frame of the simulation of plasmas flows in the occurrence of a magnetic field for ITER application in the frame of CEA and ANR program. A LRC laboratory network between CEA, Ecole Centrale Marseille and Université d'Aix-Marseille was recently constituted between a team of the laboratory and the DRFC of Cadarache on CFD modelling inside ITER research program. The list is not exhaustive and some topics are reported and illustrated within this review.

2 Aerodynamics of external and internal flows with high order methods

2.1 Ahmed body

A numerical study of the flow over the Ahmed car model (Ahmed body [2], see Fig.1 & 2), with slanted back-face 25°, is provided in collaboration with R. Pasquetti (Lab. J. Dieudonné, Nice) for a Reynolds number $Re = 768000$. It constitutes a flow at very high Reynolds number and massive separation. The study is based on a high-order large-eddy simulations (LES), carried out with a multi-domain spectral Chebyshev-Fourier solver. The LES capability is implemented thanks to a Spectral Vanishing Viscosity (SVV) technique, with particular attention to the near-wall region, and the bluff body is modeled with a pseudopenalization method. Such a SVV-LES approach is extended for the first time to an industrial three-dimensional turbulent flow over a complex geometry. A detailed analysis of the flow structures provides a better understanding of the interactions between flow separations and the dynamic behavior of the released vortex wake.

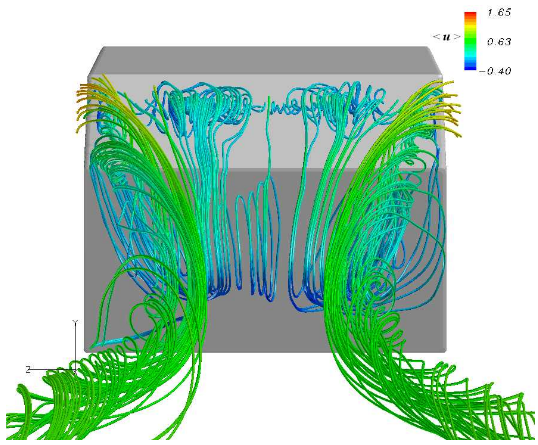


Figure 1: 3D streamline patterns colored by the longitudinal velocity $< u >$.

The complex dynamics of the wake flow is recovered in agreement with the experimental results of Lienhart et *al*. [5]. The thin turbulent boundary layer partially separates at the edge of the slant involving highly unsteady turbulent mechanisms above the slanted face and in the wake. The present SVV-LES results have shown that this partial detachment is controlled by two strong contra rotative trailing vortices (Fig.1) which interact with hairpin vortices occurring within the shear layer above the slant to form large helical structures providing strong unsteady phenomena in the wake (Fig.2). More intrinsic properties of the turbulence was pointed out as the $k^{5/3}$ energy density decay in the inertial range. Comparisons of the time averaged quantities have also shown a globally good quantitative agreement with experimental [5] measurements in the symmetry plane. An original near wall treatment based on a local relaxation of the SVV threshold has been implemented, which has considerably improved the results and particularly the production of turbulence over the slant. An improvement of the present results would come from a better prediction of the upstream flow over the roof of the body.

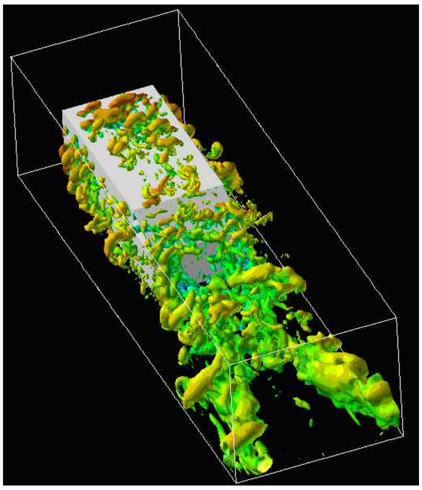


Figure 2: Iso-values of pressure fluctuations colored by $< u >$.

2.2 Rotating disk flows

Flows in rotating-disk systems are not only a subject of fundamental interest as prototype flows for investigating the underlying structures of the three-dimensional laminar and turbulent boundary layers but are also a topic of practical importance in the performance improvements of many industrial devices. Moreover, many of these flows have also features that appear in the flows in the Earth atmosphere and in wind-driven surface layers of the ocean.

Turbulent flows have been considered in an actual enclosed rotor-stator configuration with a rotating hub and a stationary shroud. Large Eddy Simulations (LES) have been performed using the Spectral Vanishing Viscosity (SVV) technique which is shown leading to stable discretizations without sacrificing the formal accuracy of the spectral approximation [15]. The numerical results have been favorably compared to velocity measurements performed at IRPHE (Marseille) for rotational Reynolds numbers up to $Re = 10^6$ in an annular cavity of large aspect ratio [14]. In the detailed picture of the flow structure that emerges, the turbulence is mainly confined in the boundary layers including in the Stewartson

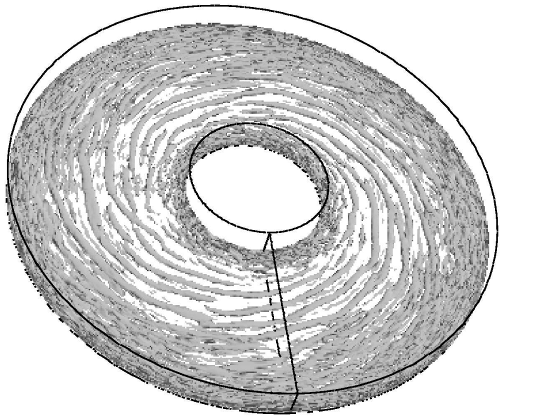
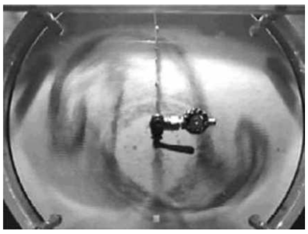


Figure 3: Isosurface plot of the Q-criterion in the rotor boundary layer for $Re = 10^6$.

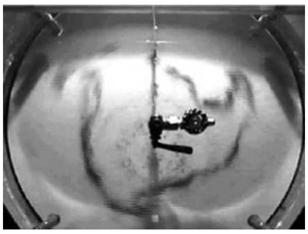
layer along the external cylinder. For Reynolds numbers $Re \geq 10^5$, the stator boundary layer is turbulent over most of the cavity. On the other hand, the rotor layer becomes progressively turbulent from the outer radial locations although the rotating hub is shown to destabilize the inner part of the boundary layers. The isosurface maps of the Q-criterion reveal that the three-dimensional spiral arms (Fig.3) observed in the unstable laminar regime evolve to more axisymmetric structures when turbulence occurs. At $Re = 10^6$, the flow is fully turbulent and the anisotropy invariant map highlights turbulence structuring, which can be either a "cigar-shaped" structuring aligned on the tangential direction or a "pancake-shaped" structuring depending on the axial location. The reduction of the structural parameter (the ratio of the magnitude of the shear stress vector to twice the turbulence kinetic energy) under the typical limit 0.15, as well as the misalignment between the shear stress vector and the mean velocity gradient vector, highlight the three-dimensional nature of both rotor and stator boundary layers with a degree of three-dimensionality much higher than in previous open systems.

These last results have been extended to the non-isothermal case. The Boussinesq approximation is used to take into account the centrifugal-buoyancy effects. The thermal effects have been examined in the same rotor-stator cavity for $Re = 10^6$ and Rayleigh numbers up to $Ra = 10^8$. These LES results provide accurate, instantaneous quantities which are of interest in understanding the physics of turbulent flows and heat transfers in such cavities. At $Ra = 10^7$, a regime with thermal plumes appear but regarding the averaged results, very small effects of density variation are obtained on the mean and turbulent fields. The radial distributions of the local Nusselt number Nu on both disks confirm previous results: Nu depends on the local Reynolds number to the power α slightly lower than 0.8 ($\alpha = 0.746$ here).

Experiments performed in collaboration with Professor B.E. Launder (MACE, Univ. Manchester) [3] have revealed the appearance at very high Reynolds numbers of three-dimensional structures in the core region of the flow in enclosed rotor-stator cavities of very large aspect ratio. These have been confirmed by recent experiments for other flow conditions. These organized vortex structures play a significant role on the entire flow - so that (U)RANS simulations can



(a)



(b)

Figure 4: Visualizations by dye injection of 3D structures in the core region of a highly turbulent rotor-stator flow: (a) S-shaped mode, (b) three-vortex mode [3].

fail to predict accurately the heat transfer in these cavities. The two-vortex S-shaped (Fig.3a) and the three-vortex (Fig.3b) patterns are stable over a wide range of conditions but higher modes have also been observed. These structures disappear when a rotating hub is attached to the stationary disk, which explains why they have not been obtained in the previous LES calculations.

3 Computational Approaches of Magnetized Plasmas Flows

Since 2005 the laboratory is involved in a research activity on numerical modelling of turbulence for tokamak fusion plasma. This activity is performed in collaboration with the "Département de Recherche sur la Fusion Contrôlée" (DRFC) of the CEA-Cadarache, in the framework of the ITER project. The ANR and LRC projects mentioned above are focused on two main research activities that are briefly described in the following.

3.1 Multiscale methods for computing fluid and plasma turbulence

The group in collaboration with M. Farge (LMD, ENS Paris) develops multiscale methods for modeling and computing fully-developped turbulent flows. The idea is to decompose the flow variables into coherent and incoherent contributions using nonlinear wavelet filtering. The coherent part is then deterministically integrated using adaptive numerical methods while the influence of the incoherent background flow is statistically modeled. The new method, called Coherent Vortex Simulation (CVS) has been successfully applied, for examples, to compute turbulent mixing layers or decaying two-dimensional turbulence [12] in a circular container [11] (Fig.5). To compute confined flows or flows around obstacles, the CVS method is associated with a voluminal penalty method. This combined approach has been successfully applied to two-dimensional flows such as the flow behind a flat wing, in a network of tubes ...

3.2 Numerical modelling of turbulence transport equations for tokamak edge plasmas

The confinement performances of tokamak plasmas are essentially governed by turbulent transport. In this framework, the transition region between closed and open magnetic flux surfaces plays a crucial role. Indeed, an edge transport barrier can develop spontaneously in its vicinity, leading to the so-called H-mode regime. This is the reference scenario for ITER, and such will focus the research effort prior and during ITER operation.

Figure 5: Decaying 2D turbulence in a circular container: vorticity field [11].

Our collaboration with the DRFC/CEA-Cadarache concerns more specifically the study of the edge region in which several open issues remain. First, there is a rich variety of non linear physical processes that are potentially important in this region, ranging from turbulence overshoot to neutral penetration. Second, the very sharp transition in this region is a challenge to many concepts used to describe transport properties. Also, the instability mechanisms are shown to change completely there. This leads to a complex boundary layer problem. Third, and not surprisingly, the non linear simulations of fluid equations require a powerful numerical treatment that is able to account simultaneously for the doubly-periodic (inner) region and the essentially non-periodic (outer) region.

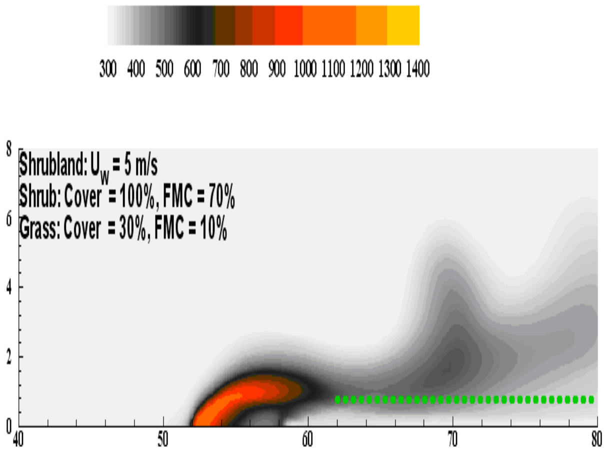
This collaboration is related to the work started with the ANR contract: M2TFP ("Développement de Méthodes Multi-échelles et spectrales pour l'analyse et la simulation numérique en Turbulence Fluide et Plasma. Application à la fusion dans les plasmas avec confinement magnétique", 2006-2008).

4 Fire modelling and simulation

For ten years, the team develops researches activity in wildfire modelling [6, 7]. They participated, in collaboration with the IUSTI laboratory in Marseille, to three European projects (EFAISTOS, FIRESTAR and EUFIRELAB) during the 4^{th} and the 5^{th} Framework Programs (FP). Actually they participate (as the main member of the Consortium) to the European project FireParadox (6^{th} FP). Inside this four years project (March 2006 - February 2010), they are in charge to develop a 3D code to simulate the behaviour of wildfires at a local scale (< 500 m) (see Fig.6a,b). This work is an extension of a 2D model that they have developed this last decade. It is based on a multiphase formulation, describing the main physical phenomena governing the propagation of a fire through a solid fuel layer. During this project, a particular effort is done to optimise and to adapt

the computational code to the multi-processor share memory machines (SGI-Altix) bought with the financial support of the European commission.

(a)



(b)

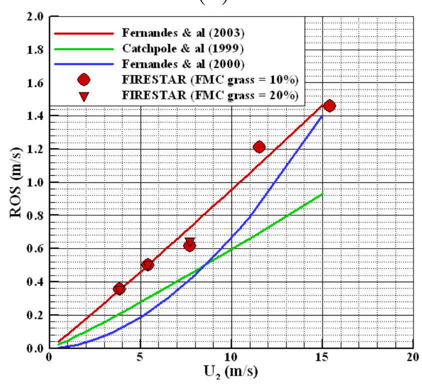


Figure 6: (a) Average temperature field calculated during the propagation of a surface fire through a Mediterranean shrubland; (b) Surface fire in a shrubland: Evolution of the rate of spread (ROS) as a function of the wind speed (numerical results compared with experimental correlations).

5 Others, international networks and web-sites

Complementary details on other activities in the MSNM-GP (L3M) laboratory are available in the web site : http://www.MSNM-GP.univ-mrs.fr/.

The CNRS-DFG Program headed in the frame of GDR Européen MFN directly involves 30 laboratories and research teams in France and Germany among them three participating to ERCOFTAC France Sud Group - that are MSNM-GP UMR 6181, Lab. J.A. Dieudonné UMR 6621 (Nice, R. Pasquetti), IMFT UMR 5502 (Toulouse, Ch. Airiau).

References on the two research topics in web sites : Generation of Noise in Turbulent Flows (http://www.iag.uni-stuttgart.de/dfg-cnrs/) and LES of Complex Flows (http://www.hy.bv.tum.de/DFG-CNRS/).

References

[1] G. Accary, and I. Raspo. A 3D finite volume method for the prediction of a supercritical fluid boyant flow in a differentially heated cavity. *Computers & Fluids*, 35 (10):1316–1331, 2006.

[2] S.R. Ahmed, and G. Ramm. Salient Features of the Time-Averaged Ground Vehicle Wake. *SAE-paper*, 40300, 1984.

[3] O. Czarny, H. Iacovides and B.E. Launder. Precessing Vortex Structures in Turbulent Flow within Rotor-Stator Disc Cavities. *Flow, Turbulence and Combustion*, 69:51–61, 2002.

[4] M. Forestier & P. Haldenwang. Analysis of high Rayleigh number thermally stratified convection that affects the vapour phase in a liquefied gas storage tank. *J. Fluid Mech.*, 588:217–241, 2007.

[5] H. Lienhart, C. Stoots, and S. Becker. Flow and turbulence structures in the wake of a simplified car model (Ahmed Body). In *DGLR Fach Symp. der AG STAB*, Stuttgart University, 15-17 November 2000.

[6] D. Morvan, and J.L. Dupuy. Modelling the propagation of a wildfire through a Mediterranean shrub using a multiphase formulation. *Combustion & Flame*, 138:199–200, 2004.

[7] D. Morvan. A numerical study of flame geometry and potential for crown fire initiation for a wildfire propagating through a shrub fuel. *Int. J. Wildland Fire*, 16:511–518, 2007.

[8] I. Raspo, S. Meradji & B. Zappoli. Heterogeneous reaction induced by the piston effect in supercritical binary mixtures. *Chem. Eng. Sci.*, 62:4182–4192, 2007.

[9] I. Raspo, C. Nicolas, E. Neau & S. Meradji. Diffusion of solids in supercritical carbon dioxide : Modelling of near critical behaviour. *Fluid Phase Equilib.*, in press, 2007.

[10] O. Roussel, and K. Schneider. Adaptive numerical simulation of pulsating planar flames for large Lewis and Zeldovich ranges. *Comm. Nonlin. Sci. Num. Simulation*, 11:463–480, 2006.

[11] K. Schneider, and M. Farge. Decaying two-dimensional turbulent flow in a circular container. *Phys. Rev. Lett.*, 95:244502, 2005.

[12] K. Schneider, M. Farge, A. Azzalini & J. Ziuber. Coherent vortex extraction and simulation of 2D isotropic turbulence. *JoT*, 7 (44):1–24, 2006.

[13] E. Serre & P. Bontoux. Vortex breakdown in a cylinder with a rotating bottom and a free surface. *Int. J. Heat Fluid Flow*, 28 (2):229–248, 2007.

[14] E. Séverac, S. Poncet, E. Serre and M.P. Chauve. Large eddy simulation and measurements of turbulent enclosed rotor-stator flows. *Phys. Fluids*, 19:085113, 2007.

[15] E. Séverac, and E. Serre. A Spectral Viscosity LES model for the simulation of turbulent flows within rotating cavities. *J. Comp. Phys.*, 226(6):1234–1255, 2007.